DETERMINATION OF VORTEX FORMATION AT TEMENGGOR TNB HYDROPOWER STATION

MUHAMAD ALIFF IKMAL BIN JAAFAR

UNIVERSITI SAINS MALAYSIA

2021

DETERMINATION OF VORTEX FORMATION AT TEMENGGOR TNB HYDROPOWER STATION

by

MUHAMAD ALIFF IKMAL BIN JAAFAR

Thesis submitted in fulfilment of the requirements
for the degree of
Bachelor of Mechanical Engineering with Honour

October 2021

ACKNOWLEDGEMENT

Before continuing further, I would like to express my gratitude to my project's supervisor, Associate Professor Dr. Mohamad Aizat Bin Abas for his continuous support, invaluable advise and patience during my bachelor's degree research. His immense knowledge and plentiful experience have encourage me in all the time of my academic research and daily life. I would also like to thank Francis Tung for his technical support on my study. Without his help, my research would not be possible. Not to forget, I would like to thank all my friends and course mate in the Mechanical Engineering Programme for their continuous emotional and technical supports. It is their kind of help and support that colour my four years of university life. Finally, I would like to express my gratitude to my family. Without them, this experiences would be an impossible journey for me to finish.

TABLE OF CONTENTS

ACKNOWLEDGEMENT ii

TABLE OF CONTENTS iii

LIST OF TABLES vii

LIST OF FIGURES viii

LIST OF EQUATION x

LIST OF SYMBOLS xii

LIST OF ABBREVIATIONS xiii

LIST OF APPENDICES xv

ABSTRAK xvi

ABSTRACT xvii

CHAPTER 1 INTRODUCTION 1

1.1 Introduction 1

1.2 Background of Study 3

1.3 Problem Statement 5

1.4 Objective of this research. 6

CHAPTER 2 LITERATURE REVIEW 7

2.1 Introduction 7

2.2 Review on the basic theory behind vortices formation 7

2.3 Review on Dimensional Analysis 8

2.4 Review on Critical Submergence Prediction 9

2.5 Review on the study of free surface vortices on Hydropower dam. 11

2.6 Review on the current usage of CFD on vortex study at hydropower intake 12

2.7 Verdict 16

2.7.1 Research gap 16

CHAPTER 3 RESEARCH METHODOLOGY 17

3.1 Introduction 17

3.2 Pre-processing 18

3.2.1 Data collection for CAD drawing 18

3.2.2 Meshing.. 22

3.2.3 Labelling on the penstock’s bending and contraction.................... 25

3.3 Simulation Setup 26

3.4 Results’ Pre-Analysis 27

3.4.1 Energy loss in the penstock... 27

3.4.1(a) Frictional loss... 27

3.4.1(b) Minor loss in penstock.. 27

3.4.1(c) Total loss.. 30

3.4.2 Pressure difference / Potential energy left for turbines. 30

CHAPTER 4 RESULTS AND DISCUSSION... 32

4.1 Vortex formation based on the simulation results... 32

4.1.1 Vortex formation at penstock 4 (20 seconds elapsed time). 32

4.1.1(a) Surface vector. 32

4.1.1(b) Surface contour 34

4.1.1(c) Vortex cross section – Velocity contour........................ 36

4.1.1(d) Vortex cross section – Volume of fraction (Surface) 38

4.2 Results validation 40

4.3 Total head loss and energy level in penstock 47

4.3.1 Water level = 244.92 m 47

4.3.1(a) Friction loss 47

4.3.1(b) Bending loss... 47

4.3.1(c) Contraction loss 47

4.3.1(d) Total loss and Energy level... 48

4.3.2 Water level = 245.53 m 48
4.3.2(a) Friction loss 48
4.3.2(b) Bending loss........ 48
4.3.2(c) Contraction loss 49
4.3.2(d) Total loss and Energy Level 49
4.3.3 Water level = 246.14 m 49
4.3.3(a) Friction loss 49
4.3.3(b) Bending loss........ 50
4.3.3(c) Contraction loss 50
4.3.3(d) Total loss and Energy level........ 50
4.3.4 Water level = 248.55 m 51
4.3.4(a) Friction loss 51
4.3.4(b) Bending loss........ 51
4.3.4(c) Contraction loss 51
4.3.4(d) Total loss and Energy Level 52
4.4 Graphical plot 53
4.4.1 Total loss in each penstock to the water level. 53
4.4.2 Energy level to the water level 54
4.4.3 Outlet flow rate to the water level 54
4.4.4 Energy level with percentage dropped on each penstock 55
4.5 Discussion of the obtained results 58
CHAPTER 5 CONCLUSION AND FUTURE RECOMMENDATION 60
5.1 Conclusion 60
5.2 Recommendations for Future Research 60
REFERENCES 61
APPENDICES
LIST OF PUBLICATIONS

LIST OF TABLES

Page

Table 2.1 - Equation for Critical Submergence Prediction 10

Table 3.1 Gradual contraction loss 29

Table 4.1 Surface vector at the intake 32

Table 4.2 Surface contour at intake 34

Table 4.3 Velocity contour of vortex cross section 36

Table 4.4 VOF contour of vortex cross section 38

Table 4.5 Friction loss for all penstock 47

Table 4.6 Bending loss in all penstocks at 244.92 m 47

Table 4.7 Contraction loss in all penstocks at 245.92 m 47

Table 4.8 Total loss and energy level at 244.92 m 48

Table 4.9 Friction loss for all penstock 48

Table 4.10 Bending loss in all penstocks at 245.53 m 48

Table 4.11 Contraction loss in all penstocks at 245.53 m 49

Table 4.12 Total loss and energy level at 245.53 m 49

Table 4.13 Friction loss for all penstock 49

Table 4.14 Bending loss in all penstocks at 246.14 m 50

Table 4.15 Contraction loss in all penstocks at 246.14 m 50

Table 4.16 Total loss and energy level at 246.14 m 50

Table 4.17 Friction loss for all penstock 51

Table 4.18 Bending loss in all penstocks at 248.55 m 51

Table 4.19 Contraction loss in all penstocks at 248.55 m 51

Table 4.20 Total loss and energy level at 248.55 m 52

LIST OF FIGURES

Page

Figure 1.1 - Primary energy supply in Malaysia (2018) 2

Figure 1.2 - Energy sources over time (2018)..3

Figure 2.1 - Hecker classification of vortex [9] 8

Figure 2.2 Side by side comparisons between simulation and experiment [20]. 13

Figure 2.3 Example of grid independence test by comparing maximum air VOF and grid's number. 15

Figure 3.1 Satellite view of Temenggor TNB Hydropower dam 18

Figure 3.2 Satellite view of Temenggor’s dam intake and outlet (green box). ...19

Figure 3.3 CAD model of the Temenggor dam and TNB hydropower station. ..19

Figure 3.4 Temenggor dam with wireframe view of the soil 20

Figure 3.5 Intake view of the Temenggor dam. 20

Figure 3.6 Inlet for the simulation (Highlighted) 21

Figure 3.7 The penstock of Temenggor dam without the soil enclosure. 21

Figure 3.8 Grid independence test results from varying element length............24

Figure 3.9 Tetrahedron mesh of the Temenggor dam. 25

Figure 3.10 Bending and Contraction labelling..25

Figure 3.11 VOF of Temenggor intake with 244.92 m free surface level. 26

Figure 3.12 - Gradual Contraction Transition..28

Figure 3.13 Two points for energy equation 31

Figure 4.1 Real average water discharge rate at Temenggor hydropower station per day 40

Figure 4.2 Area of the discharge outlet 41

Figure 4.3 Total loss on each penstock vs water level 53

Figure 4.4 Energy level according to corresponding water level........................54
Figure 4.5 Flow rate vs the water level 54
Figure 4.6 Energy level and percentage dropped in penstock 1..........................55
Figure 4.7 Energy level and percentage dropped in penstock 2..........................56
Figure 4.8 Energy level and percentage dropped in penstock 3..........................57
Figure 4.9 Energy level and percentage dropped in penstock 4..........................58

LIST OF EQUATION

Page

Equation 2.1Critical submergence function parameters 9

Equation 2.2 Relative critical submergence function parameters...............................9

Equation 2.3 Submergence enveloped region (Gordon, 1970).................................10

Equation 2.4 Lower band for S/D ratio..10

Equation 2.5 Upper band for S/D ratio 10

Equation 3.1 Continuity equation in space in cartesian coordinates..........................17

Equation 3.2 Navier-Stoke equation in x-direction...17

Equation 3.3 Navier-Stoke equation in y-direction...17

Equation 3.4 Navier-Stoke equation in z-direction...17

Equation 3.5 Nonlinear partial differential equation...17

Equation 3.6 Free surface..18

Equation 3.7 Friction loss 27

Equation 3.8 Specialised friction factor equation for turbulent flow.........................27

Equation 3.9 Minor loss due to bending 28

Equation 3.10 Loss coefficient for sharp bending 28

Equation 3.11 Loss coefficient due to gradual contraction.......................................28

Equation 3.12 Angle of contraction 28

Equation 3.13 Head loss due to gradual contraction...28

Equation 3.14 Total head loss...30

Equation 3.15 Pressure difference between two points 30

Equation 3.16 Simplified Bernoulli equation...30

Equation 3.17Net head differences 31

Equation 4.1 Flow rate calculation...41

LIST OF SYMBOLS

F_r	Froude Number
R_e	Reynolds number
We	Weber Number
Γ	Vorticity
S	Submergence
S_c	Critical Submergence
S_{rc}	Relative Critical Submergence

LIST OF ABBREVIATIONS

C2ES	Centre for Climate and Energy Solutions
TNB	Tenaga Nasional Berhad
LIDAR	Light Detection and Ranging
CAD	Computer Aided Design
FSI	Fluid-Structure Interaction
CFD	Computational Fluid Dynamics
VOF	Volume Of Fraction
LES	Large Eddy Simulation
SST	Shear Stress Transport
BC	Boundary Conditions

DETERMINATION OF VORTEX FORMATION AT TEMENGGOR TNB HYDROPOWER STATION

LIST OF APPENDICES

Appendix A Simulation results

PENENTUAN FORMASI VORTEX DI STESEN PENJANA KUASA TENAGA HIDROTEMENGGOR TNB

ABSTRAK

Tenaga elektrik yang dihasilkan oleh Stesyen Penjana Hidroelektrik Temenggor hanya menghasilkan 80% daripada kapasiti keseluruhan. Hal ini disebabkan oleh pembentukan pusaran air di tempat takungan air (Tasik Temenggor). Mod kajian ini memfokuskan kepada simulasi berangka menggunakan perisian ANSYS fluent. Simulasi ini dimodelkan dengan menggunakan model pergolakan k-epsilon and digambarkan dengan kaedah 'Volume of Fraction' (VOF) pada paras air yang berbeza untuk memerhati pusaran air yang terbentuk pada pengambilan hidraulik. Pengesahan hasil kajian pula dilakukan dengan membandingkan keputusan kadar aliran simulasi dan kadar aliran sebenar. Analisis pula dijalankan dengan membandingkan tiga pembolehubah yang diukur iaitu tahap tenaga, kadar aliran air, dan tahap tenaga yang hilang kepada pembolehubah yang diubah iaitu kedalaman air. Berdasarkan keputusan kajian, pusaran air terbukti mengganggu aliran pada mengambilan hidraulik. Disebabkan oleh, pusaran yang selalu terbentuk pada setiap paras air, penstock 4 merupakan penstock yang paling terjejas.

DETERMINATION OF VORTEX FORMATION AT TEMENGGOR TNB HYDROPOWER STATION

ABSTRACT

The power generated by Temenggor Hydropower station is only 80% of the station's full capacity. The reason behind this reduced power generation, is due to the formation of free surface vortices at the hydraulic intake. This study is focuses on numerical simulation using ANSYS fluent software. The model was built using SolidWorks and the data were obtained from the dam's blueprints and LIDAR scanner (Light Detection and Ranging) for the intake topology. The mesh sensitivity analysis was then carried out by testing different element sizes. This study was carried out by simulating the dam water flow using k-epsilon turbulence model and volume of fraction (VOF) at different water level in order to observe the vortices formation at hydraulic intake. The results validation is then carried out by comparing the simulation outlet discharge with the real discharge. The analysis was carried out by comparing the energy level, outlet flow rate and total loss in penstock with respects to different water level. The findings from the simulations and results are, the vortices disrupt the intake flow but with increasing water level the effect reduced. Penstock 4 are affected the most due to vortices consistently form at any water level.

CHAPTER 1

INTRODUCTION

1.1 Introduction

Energy is defined by the ability or capacity to do work [1]. Energy is very important factor to the growth of civilisation. It enables us to move around faster, to communicate between long distances and to lit up a room. As we know, energy cannot be destroyed nor created, because of this, humans adapt and learn to change and convert the energy from one form to another. For example, electric energy was harnessed by converting potential energy (hydropower) or chemical energy (coal) to mechanical energy (turbine). The most popular energy sources globally are chemical energy. This energy is mostly obtained from fossil fuel which fall under non-renewable category. This type of sources impacting our environment negatively. It causes climate change due to high carbon emission. The released carbon traps heat in the atmosphere which then causes nominal temperature of earth to climb. As a result, climate change happens. Because of this, the developer start questioning the sustainability issue on their design. The solution is simply by using renewable source such as solar, wind and hydropower. Based on article from C2ES (Centre for Climate and Energy Solutions), renewable energy is the fastest growing energy source in the world and this number will keep rising [2]. Renewable energy source is free and does not emit carbon, but why most of the energy source came from fossil fuel?

Renewable energy is difficult to implement. Besides having costly maintenance and intermittency problem, they are not-so-efficient as compared to fossil fuel [3]. As stated earlier, energy can only be converted from one form to another. But what people do not realise; the energy is not 100% converted to another form. With each conversion, some energy will losses to the surrounding into another unintended forms [4]. And if the conversion step is high, the energy losses is also high. Optimisation on each component is crucial to ensure high efficiency. In conclusion, it is very important for renewable energy to have highly efficient system.

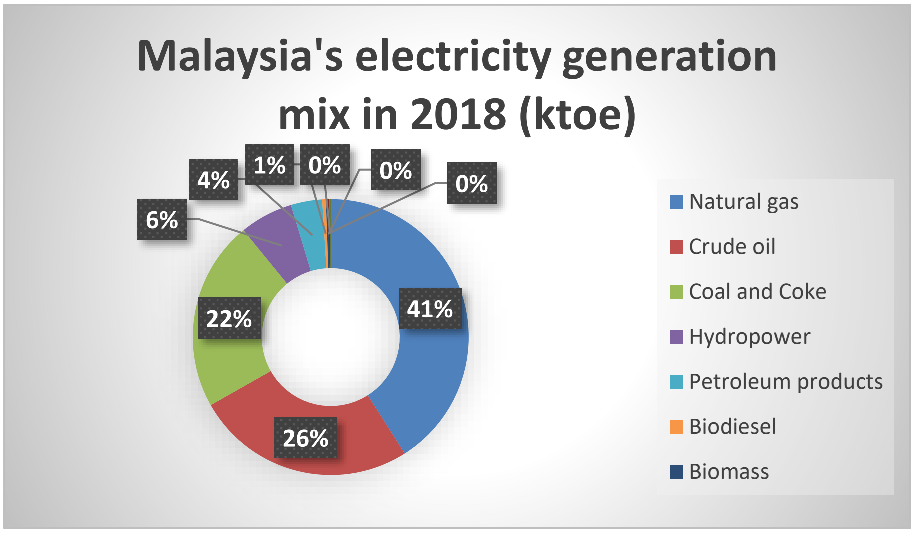


Figure 1.1 - Primary energy supply in Malaysia (2018)

Hydropower source are popular renewable energy in Malaysia [5]. The hydropower dam converts gravitational potential energy to mechanical energy to spinning the turbine. The reason on why it is so popular in Malaysia is that we have good geographical factor for hydropower implementations [6]. The country has lots of mountain valley that can be used to trap water. Not only that, the shape of the river also bring potential to build small run-of-river hydropower [6]. These sources are managed by Tenaga Nasional Berhad (TNB).

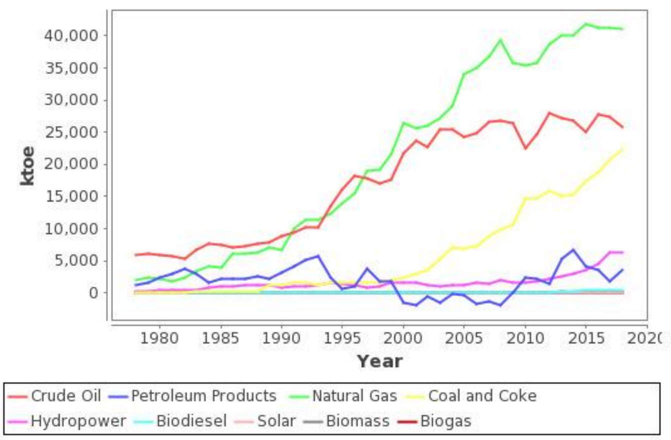


Figure 1.2 - Energy sources over time (2018)

The natural gas contributes the most on power generation in Malaysia at around 41% of total energy production (40,939 ktoe). Followed by Crude oil (26%, 25,771 ktoe), Coal and Coke (22%, 22,280 ktoe) and Hydropower (6%, 6230 ktoe). From figure 1, the number of hydropower generation is not that significant. But if we look the trend of the energy sources over the years on figure 2, the hydropower share maintains while the top two (Natural gas and Crude oil are declining). That being said, the coal and coke share are rapidly increasing. This source are also a type of fossil fuel. As we know, fossil fuel can cause great problem to our environment. As an example, air pollution from the incomplete combustion of the fuel. This releases carbon monoxide and soot (carbon) which can affect the life if it were inhaled. So, it is important for us to use cleaner energy, to maintain sustainability to the future generations.

1.2 Background of Study

Tenaga Nasional Berhad (TNB) are the main organisations that manage electricity sources in Malaysia. Recently, TNB found a huge problem regarding the amount of power generated on one of their hydropower stations; namely Temenggor hydropower station. The power generated is only 80% of the station's full capacity. The reason behind the reduced power generation, is due to the formation of free surface vortices at the water reservoir. As mentioned earlier, renewable energy source is not-so-efficient and have costly maintenance, so, this is not good since the reduced power generation can cause losses of money to the TNB. This study is important because the need to identify how the vortices formation cause problems to the station performance and overall structures.

This study will be beneficial to the community, especially to the people that live in the northern Malaysia. This because, Temenggor hydropower station supply electricity to the northern part of Malaysia. Not only limited to the general community, but this research will also give benefit to the engineering fields. Vortices formation is still a mystery phenomenon that yet to be understand to the scientist and engineers. Most of the research papers that was published have different opinions on how these phenomena might have occurred.

When the effect and the causes of vortices formation are known, it can be used to advantages to take preventive measure on the dam structure. This preventive measure includes anti-vortex devices or even changing the topology of the water reservoir. Vortices tend to entrain the debris from the bottom into the intake, if the magnitude is too strong.

This project will be done by drawing the geometry of the dam. The information of the topology of the water reservoir bottom was collected on site by using drone that are equipped with LIDAR scanner. Not all area was drawn, only the intake area that was confirmed to have vortices formation. This is to increase computational time during simulations.

The topology then will be drawn together with the dam by using CAD software called 'SolidWorks'. Normally the data obtained from the LIDAR scanner can be converted into CAD files. But the conversion can take lots of time and computational resources. Since we are drawing from scratch, the drawing needs to be accurate as possible. The completed CAD files will be loaded into ANSYS. This will be run through simulations module; Fluent (flow).

The simulation results are expected to show us where is the location of the vortices formation. Furthermore, we can also see the impact of the vortices to the intake structure according to the magnitude of the vortices and the dam operating conditions.

1.3 Problem Statement

Free surface vortices are a serious problem in hydropower dam facilities. It can pull air from water surfaces into the inlet, disrupting water flow which then cause low water intake to spin the turbine. Because of this reduced efficiency of the dam, Tenaga Nasional Berhad also lose lots of money each day. The problem become more serious when water hammer happens, especially after hydraulics jumps [7]. Water hammer is phenomenon where the sudden stops of the water flow create shockwave to the wall of the pipe, this creates 'hammer' effect to the wall. Even more, the air mixed with water will separate during water hammer. This creates voids in pipe, which in turn cause corrosion to the pipe. This accelerates damages to the pipe. If this happens frequently, the dam structure might be in danger and cost someone's life. In this project, we are going to determine the effect of free vortex formation on the hydropower dam using Ansys Fluent (Flow). Most of the research and simulation on the hydropower dam uses experimental method by building small scale dam.

Simulation using CFD are proven to be cost and time effective than building a scaled-down version of the dam. Besides, it does not interfere with simultaneous scaling of Reynolds and Froude number as we can simulate the dam at full scale.

1.4 Objective of this research.

1. To investigate the effect of intake's water level on the discharge outlet flow rate.
2. To validate the simulation with data measured from the site.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Vortices formations are mysterious field that yet to be understandable by the researchers. Even though the vortices formation studies are extensive and have been done by many, the opinions between researchers are still divided. Here is the summary of accumulated research papers that have been published about vortices formation.

2.2 Review on the basic theory behind vortices formation

Vortex behaviours are governed by the vortex strength. This strength in turn, are controlled by the circulation, $\Gamma = 2nrV\theta.$ Γ are affected by the relative submergence S/D, Froude number, Fr, and the geometrical shape of the water reservoir [8].

Hecker classify vortex into six classes [9]. This classification is based on the vortex's strength and evolution. Furthermore, the air core intensity increases, from type 1 to 6. The illustration of these classification is as in figure 1. Then Moller (2015) introduces additional class to the Hecker’s list named VT0 (type 0) [10]. Type 0 basically state that there is no activity happens at this stage.

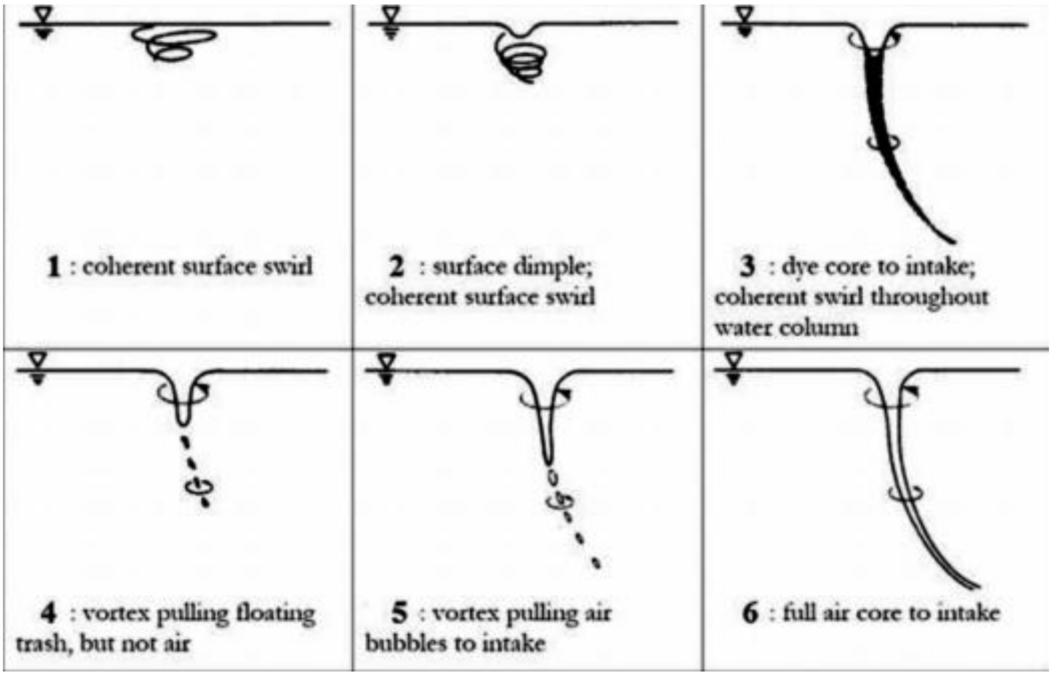


Figure 2.1 - Hecker classification of vortex [9]

Sarkardeh (2010) on the other hand categorises the vortex into three classes [11]. Class A, B and C with A being strongest, and C being the weakest.

1. **Class A** vortex is where the air is entrained into the intakes (bubble, or stable air core for worst case scenario)
2. **Class B** vortex does not entrained air, it has extended rotation downward, this enables the possibility of this vortex to pull the debris from the surfaces.
3. **Class C** vortex is considered to be safe, and it is connected with modest rotation and a little indentation on the water's surface.

2.3 Review on Dimensional Analysis

As mentioned earlier, the study of vortices at hydraulic intake is not yet to be understand by researchers.

Submergence, S was defined by Knauss as distance between intake axis and free surface of the reservoir [8]. Critical submergence., S_c is defined as shown in the equation below, the depth at which the air-core vortex forms at the intake is a function of a number of variables [8].

$$S_c = f(V, D, L, \Gamma, g, \rho, \sigma, \mu) \qquad \text{Equation 2.1}$$

Dimensional analysis of equation above resulting equation for relative critical submergence, Src [12],[11],[13].

$$S_{rc} = f\left(\frac{L}{D}, \frac{\Gamma}{VD}, F_r, R_e, W_e\right) \qquad \text{Equation 2.2}$$

The dimensionless number, Froude number, $F_r = V/(gD)^{0.5}$, Reynold number, $R_e = VD/\upsilon$, Weber number, $W_e = V(\rho D/\sigma)^{0.5}$ and other dimensionless group in the equations are important parameters. But the most prominent is the usage of Froude number, Fr, it is considered as the important parameter for vortex intensity [14].

2.4 Review on Critical Submergence Prediction

From Domfeh et al. [8], definition of the critical submergence, S_c is differ from one researchers to another. According to Jain et al. [14], it represents the minimum depth required to avoid the creation of powerful and troublesome vortices. Naderi et al. [15] on the other hand considers it to be the submerged depth between the free surface and the intake where the air entrained vortex can be seen clearly. Odgaard [16] specified S_c as the depth at which the air-core vortex's tip just makes contact with the intake. Sarkardeh et al. [11] sees critical submergence as the lowest depth below which air-core vortices cannot develop.

A number of recommendation was proposed by Prosser [17], for the intake design and the submergence specification at the intake. For horizontal intake, the study indicated 1.5D submergence (D is the diameter of the intake). He also emphasised the need for a hydraulic model review if his proposal is not accurate [17].

The earliest study of intake design was developed by Gordon (1970) at stated by Domfeh et al. [8]. The study provided an enveloped region.

$$S = CVD^{0.5} \qquad \text{Equation 2.3}$$

Where the coefficient C is 0.3 and 0.4 for symmetrical and asymmetrical flow approach respectively [8]. The generic applicability of this guideline, in the sense that the parameters utilised were not dimensionless, is a fundamental flaw [8].

A dimensionless intake design was developed by Gordon (1970), and Reddy and Pickford (1972), where the equations below refer to upper and lower bands respectively [8].

$$\frac{S}{D} = Fr \qquad \text{Equation 2.4}$$

$$\frac{S}{D} = Fr + 1 \qquad \text{Equation 2.5}$$

The observation on above the line described by Lower band, shows that the majority of intakes have free surface vortices. This indicate that the critical submergence is always greater than Fr. The observation also reveals that the critical submerged data was discovered to be located between the top and lower bands. The equations are only valid if there are no swirl-inducing structures upstream.

The other equation is also used to predict the critical submergence:

Table 2.1 - Equation for Critical Submergence Prediction

Author	**Equations**
Gordon	$\frac{S_c}{D} = 2.3 \times F_r$

Amphlett	$\frac{S_c}{D} = c \times F_r{}^{0.5} \times -0.5$
Knauss	$\frac{\boldsymbol{S_c}}{\boldsymbol{D}} = \begin{cases} \mathbf{1.5}, & \boldsymbol{Fr} < \mathbf{0.5} \\ \mathbf{2} \times \boldsymbol{Fr} + \mathbf{0.5}, & \boldsymbol{Fr} > \mathbf{0.5} \end{cases}$
Sarkardeh et al.	$\left(\frac{\boldsymbol{S_c}}{\boldsymbol{D}}\right)_{\boldsymbol{A}} = \mathbf{2}\left(\frac{\mathbf{1}}{\boldsymbol{z}}\right)^{\mathbf{0.008}} \boldsymbol{F_r}{}^{\mathbf{0.334}}$
Denny and Young	$\frac{\boldsymbol{S_c}}{\boldsymbol{D}} = \mathbf{0.151} + \mathbf{0.305}\boldsymbol{v} \times \mathbf{0.01}\boldsymbol{v}^{\mathbf{2}}$
Nagarkar et al.	$\frac{\boldsymbol{S_c}}{\boldsymbol{D}} = \mathbf{4.4} + \boldsymbol{v}^{\mathbf{0.54}} \times \boldsymbol{D}^{-\mathbf{0.73}}$

2.5 Review on the study of free surface vortices on Hydropower dam.

Free surface vortices are frequently occur when the water reservoir level are too low. The researchers found out that the low water level causes low inflows which is the main factor that causes vortices [8]. This formation can cause disturbance to the hydropower plant. Normally, the turbine are designed to take smooth flow from free surface to pressurized penstock [18]. But the flow become unpredictable when the vortex forms. This, of course are not desirable because the turbine are not designed to take swirling flow [19].

The study of Karun dam III uses physical modelling to study the vortices formation at its hydraulic intake. The model was built at optimum scale so that it minimized that the scaling effects. The scale of the constructed models is 1:33.33. This study are focuses on measuring the velocity profile at the intake, with this, flow descriptions are obtained. The velocity is measured by two different elevations. This means, that this paper are focuses on the effect of submergence to the velocity profile. The first part of the experiment is estimation/prediction of critical submerge depth, S_{cr} by using empirical equation and then validated via experiment.

Some equation such as Denny and Young (1957) and Sarkardeh et al. [11] have better prediction to the experimental results. It is also shown that at S/D = 1.6, Sarkardeh's class A vortex formed. Increasing the submerged depth make the maximum velocity plane to be reduced by 30%. Despite this when we reversed the submerged depth, the maximum plane velocity decreased about 40%.

2.6 Review on the current usage of CFD on vortex study at hydropower intake

Traditionally, experimental and analytical methods have been used to examine vortices formations. For experimental method, the geometrical setup was usually scaled down, while preserving same physical phenomena as the one happens at real scale, this is also known as similitude. However, to performs this, it can be labour intensive and expensive.

Based on 2020 review paper, scaling effects can also cause limitation to these methods. This because, full attention were needed on the physical ratio such as gravity effect between prototype and model [8]. This will affect the scaling of Reynolds, Froude and Weber number (As mentioned earlier the three main parameters for critical relative submergence).

On another subject, vortex formation is site specific, so analytical approaches are too impractical to follow. Because of this, numerical simulation that uses CFD software like ANSYS Fluent and MATLAB's CFDTool are still being used among researcher globally just because it faster and relatively inexpensive. Commercial CFD software capable to produce positive results for specific conditions of simulations. Not only that, specific area of interest can be analysed and study in CFD based [20]. Because a model can be simulated at full physical scale, CFD does not have issues with the scaling problem, thus, there was no issues arises due to scaling problems. Most of the published studies that uses numerical simulation have experimental setup to functions as a validation to the simulation results. This because, the simulation's results sometimes deviate from the experiment's results.

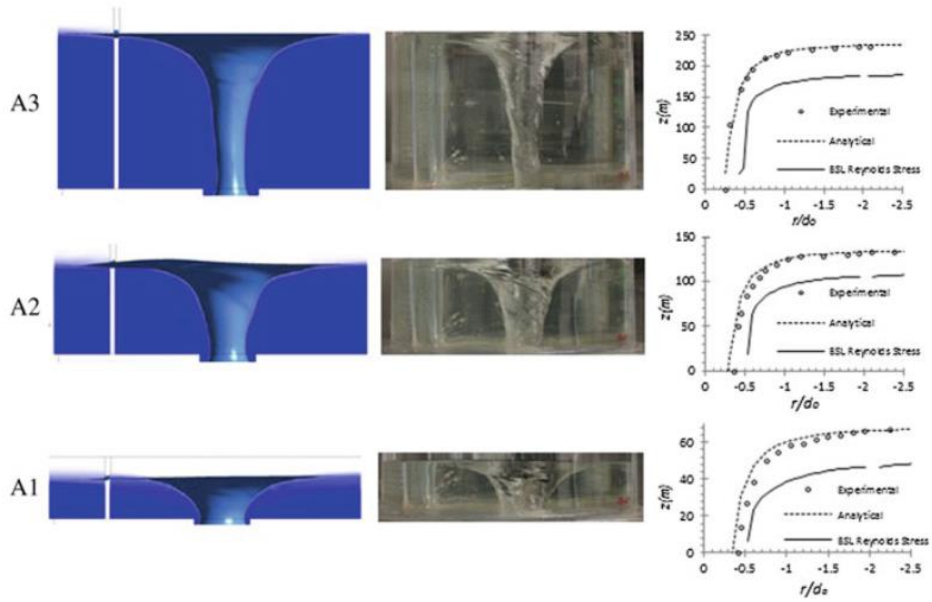


Figure 2.2 Side by side comparisons between simulation and experiment [20].

Generally, in CFD, the governing equation (Navier-Stokes Equation) was used in solving the fluid movement in its discretized state, as a result, its provides spatial and time-dependant solution [21]. The impact of surface tension, gravity, buoyancy, and density change between liquid and gas accounts for most of the complexity of flow properties including free surface phenomena.[22]. Volume of Fluid (VOF) approach are used generally by researchers to simulate the surface of the water [22]. The concept behind this approach is for each cell, they are assigned between value 1 and 0, where this value refers to water and air, respectively. If the cells are partially filled between the substances it will be assigned with an integers between 1 and 0 [23].

After then, a simple transport equation for this phase fraction α is used to estimate the evolution of the interface site. Meanwhile, Rabe et al. [24], Rabe et al. [23], Sarkardeh [25] and Sarkardeh et al. [26] recommended the use of Large Eddy Simulation (LES), because of its unique feature of using a spatial filtration process, which allows it to explicitly simulate large scale vortices in the flow, it is was recommended for turbulence modelling of air-core vortices at power intakes. The Shear Stress Transport (SST) model has also been shown to be a good turbulence model for free surface vortices simulation [20], [27]. The Reynolds stress model, however, beat the SST k-model in a comparison study including multiple turbulent models to simulate strong air-core vortices, even though the SST k-model is regarded resilient and computationally economical in comparison to the Reynolds stress model. [20].

Despite the advantages of these methods over one another, it is all depended on mesh sensitivity analysis. This analysis method is different with each paper. Based on the results, this can yield different simulation setup based on the goal of the study. In 2016, study of tidal wave station, the method was simply comparing the maximum air volume fraction with grid's number (as in Figure 2.3) [22]. Grid independence test were widely used to test the mesh sensitivity. This test was done by running the simulation for various mesh parameters and see the results deviations. For instance, grid independence test was done by comparing structured (hexahedral) and unstructured (tetrahedral) with various difference in parameters like density level, number of nodes, and number of elements [28] .

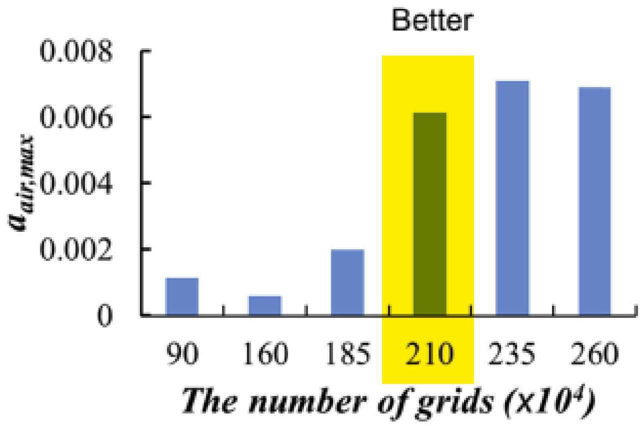


Figure 2.3 Example of grid independence test by comparing maximum air VOF and grid's number.

Aside from that, the tool that was used was able to simplify the complete process of free surface vortices evolution at intakes. [25]. Similarly, in a study by Ahn et al. [29], A free surface vortex created at a tidal power plant's intake was numerically simulated and validated, allowing for a better understanding of the vortex's effects under various operating situations. CFD techniques have also been shown to be effective at simulating the velocity flow environment surrounding the creation of free surface vortices, as well as reproducing the spiral motion related with the formation of free surface vortices [26]. Rabe et al. [23] performed a numerical study of the experimental work of Hite and Mih [30] with the results of the experimental investigation and other analytical models were found to be in good agreement in terms of radial velocity, tangential velocity, and water surface profile using FLOW-3D. Sarkardeh [25] found a strong connection between the results of his numerical simulation and the outcomes of his experimental study of Möller et al. [10] when it comes to quantifying air-entrainment rates. An interesting finding from his study is that the critical submergence could be reduced to about 12% if an air entrainment rate ratio β = 1 × 10-5 is permitted. It was also evident from the literature survey that all the numerical studies utilised commercial CFD codes such as FLOW- 3D [13],[14],[15],[16] and ANSYS CFX [22],[20].

2.7 Verdict

The contribution of vortices formation in recent year are mostly done via numerical simulation using CFD tools. However, I found out that this area of studied are yet fully understood, despite having done many physical setups. The usage of CFD tools is yet to be utilize by the researchers to study this field. Based on my review, the amount of published paper that use CFD is not abundant as compared to experimental method. Furthermore, numerical simulation needs to be validated through experimental setup. Nowadays, computational power is expected to be increased exponentially each year with so many innovations in silicon manufacturing. With this, Domfeh et al. [8] predicts that CFD tools development in recent years are so advance that it is reliable to be used in hydropower design.

2.7.1 Research gap

Most of study surrounding vortices are mostly done via experiments. Most of the published papers are focusing on the predicting the critical submergence, an important parameter where vortices formation is likely to occur. In this paper, we will be focusing more on the how the vortex formation negatively impacts the water flow to the turbine, where it directly affects the power generation performance. A similar study related to this paper is the study of Karun Dam III. It uses physical modelling to observe formation of the vortex to study the velocity profile at the intake. Unlike this paper, we will study how the disturbance of water flow due to vortex impacting the power generation.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Introduction

There is two governing equation that are used in CFD application for constant density, namely continuity equation and Navier-Stokes's equation. Both equations describe the conservation of mass and momentum, respectively.

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad \text{Equation 3.1}$$

Navier-Stoke equation:

$$\frac{1}{\rho}\frac{\partial p_d}{\partial x}=-\frac{Du}{Dt}+\mu\nabla^2 u \qquad \text{Equation 3.2}$$

$$\frac{1}{\rho}\frac{\partial p_d}{\partial y}=-\frac{Du}{Dt}+\mu\nabla^2 u \qquad \text{Equation 3.3}$$

$$\frac{1}{\rho}\frac{\partial p_d}{\partial z}=-\frac{Du}{Dt}+\mu\nabla^2 u \qquad \text{Equation 3.4}$$

Where $p_d=\rho gh$, a constant hydrostatic condition due to gravity, g; $\nabla^2$is Laplace operator, and nonlinear partial differential equations:

$$\frac{Du}{Dz}=\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z} \qquad \text{Equation 3.5}$$

This equation is discretised in space and time and solvable in various numerical scheme. This equation is often simplified due to complexity of turbulence. In practise, the renormalised Group k-energy dissipation equation has always been a success for vortices modelling [31]. The primary variables for evaluation in these equations are velocity, fluid percentage, pressure, and temperature.

The free surface equation is as follow:

$$y_{surface} = y_{free} + \frac{v^2}{2g} \qquad \text{Equation 3.6}$$

Where, y_{total} is total height, y_{free} is free surface level, v is velocity and g is gravity.

3.2 Pre-processing

In this section, model was defined for the simulation input. Solidworks was used to draw the Temenggor dam model.

3.2.1 Data collection for CAD drawing

The data collection was done by scanning the bottom of the lake using a drone equip with LIDAR scanner. By doing this, we will get an accurate topological mapping of the hydraulic intake. This step is important as the vortex's formation are highly influenced by the geological shape.

Figure 3.1 Satellite view of Temenggor TNB Hydropower dam

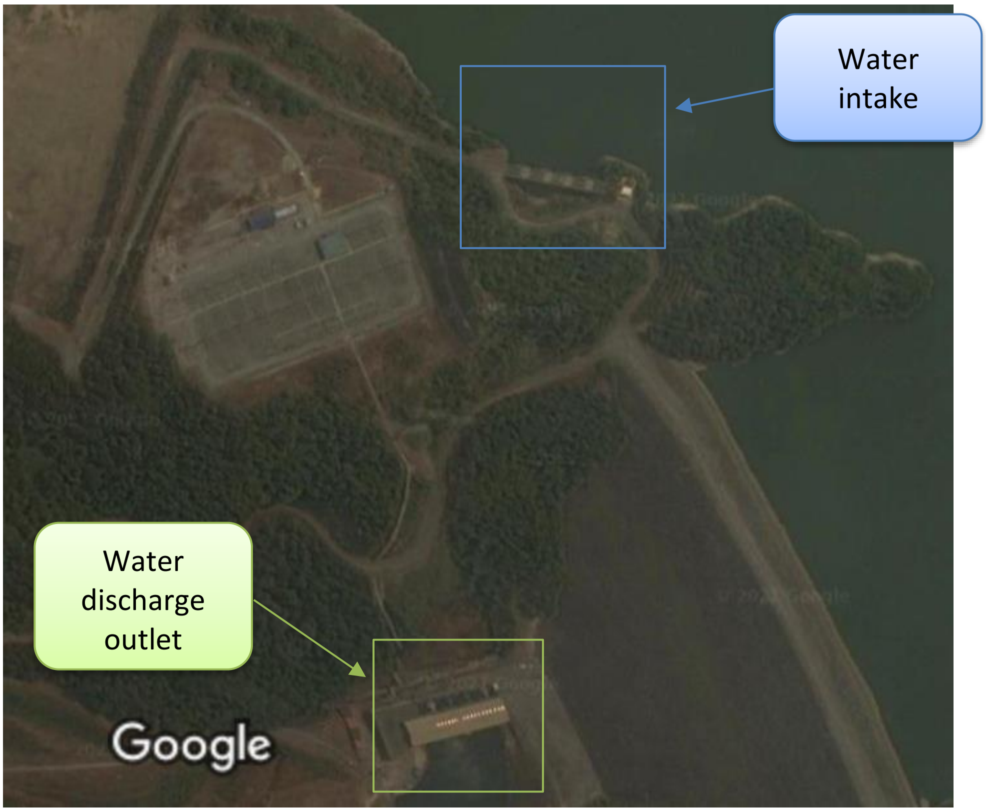


Figure 3.2 Satellite view of Temenggor's dam intake and outlet (green box).

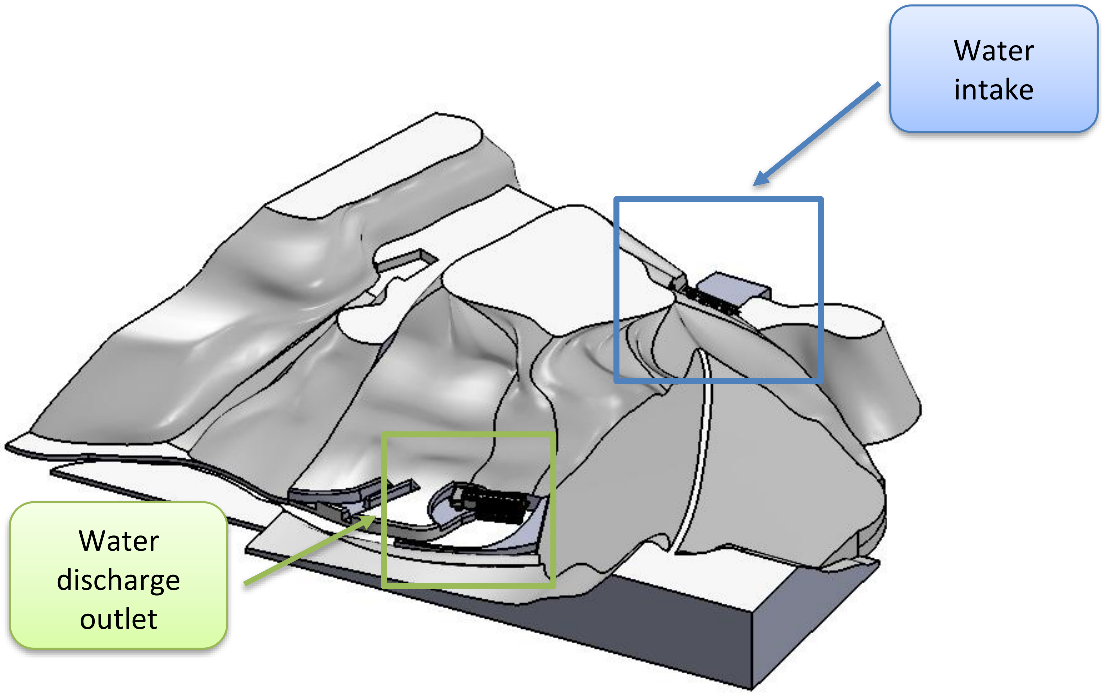


Figure 3.3 CAD model of the Temenggor dam and TNB hydropower station.

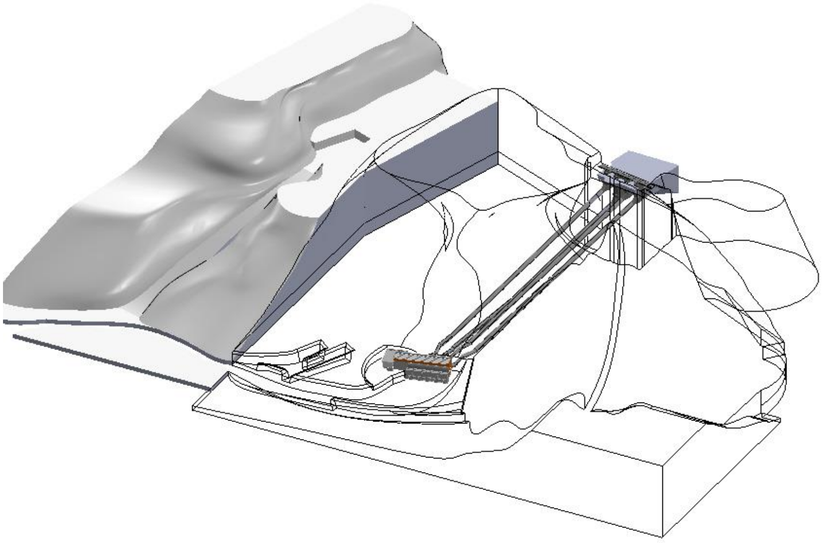


Figure 3.4 Temenggor dam with wireframe view of the soil

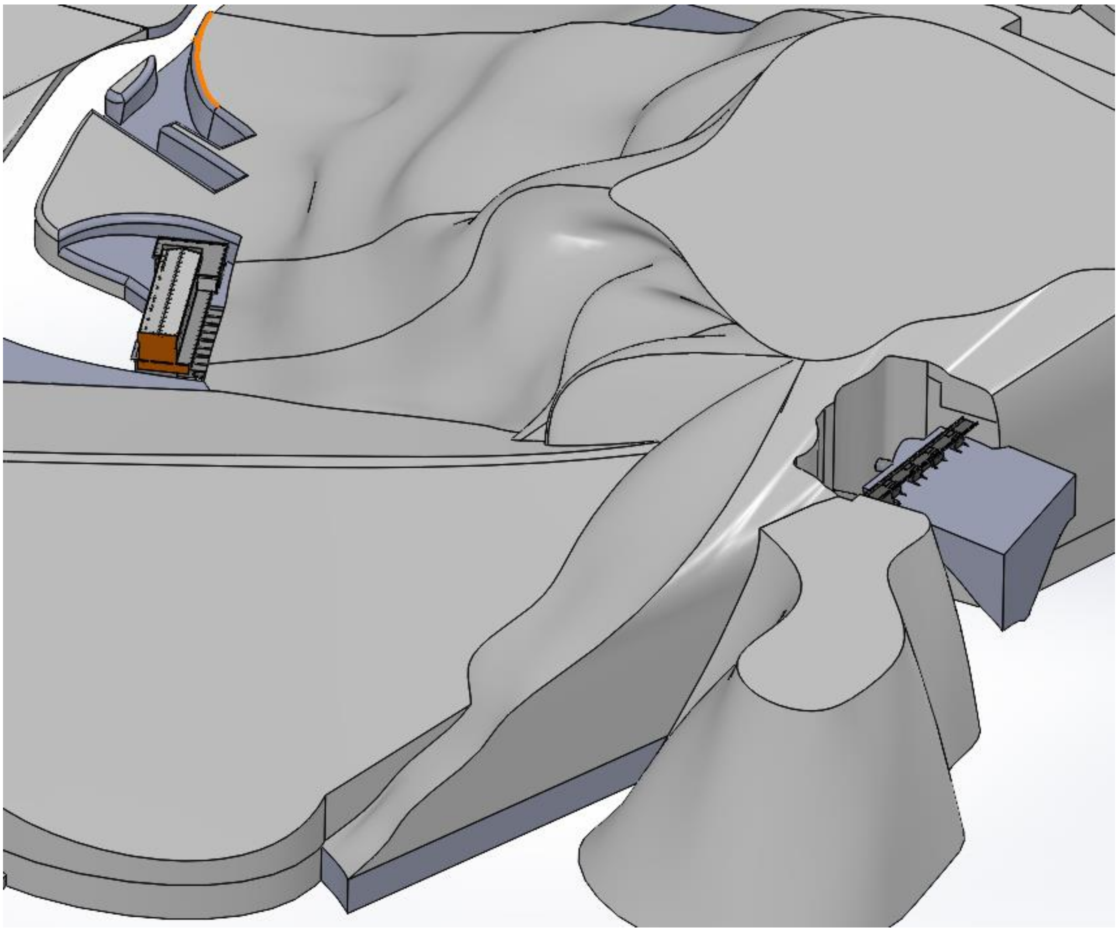


Figure 3.5 Intake view of the Temenggor dam.

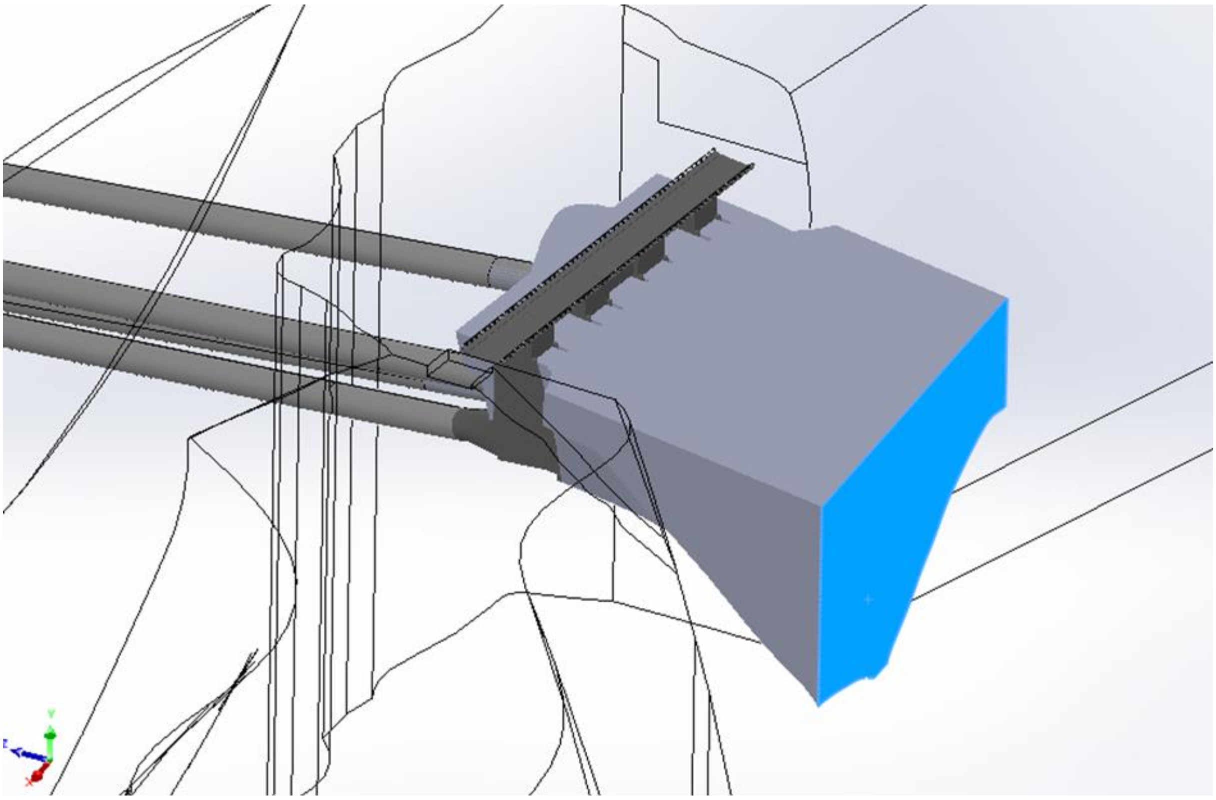


Figure 3.6 Inlet for the simulation (Highlighted)

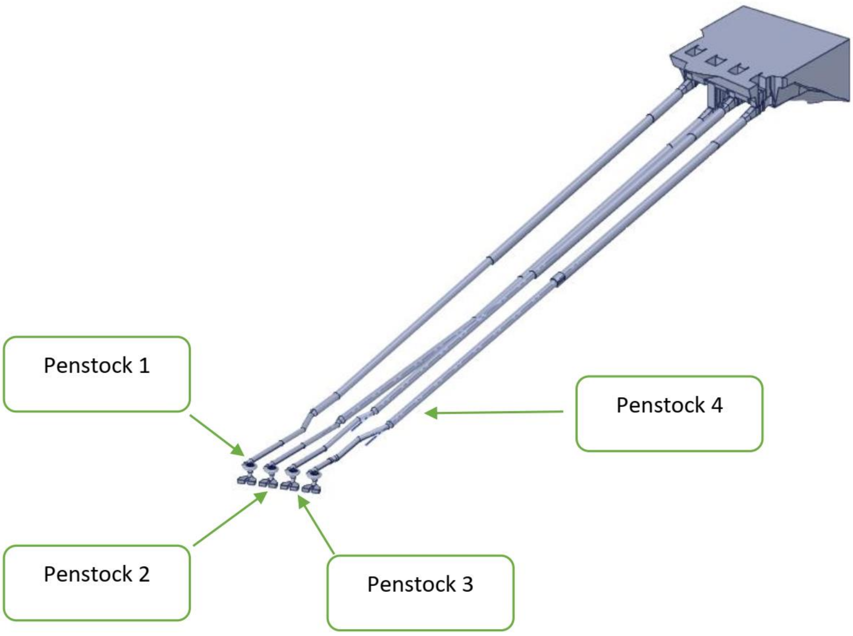


Figure 3.7 The penstock of Temenggor dam without the soil enclosure.

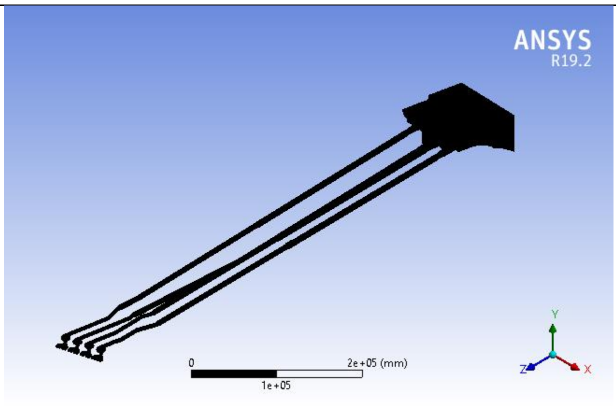
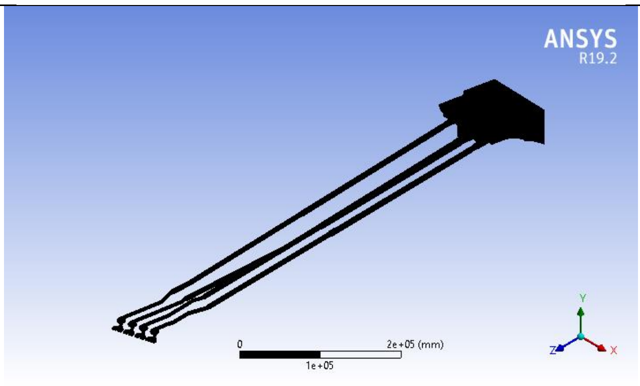
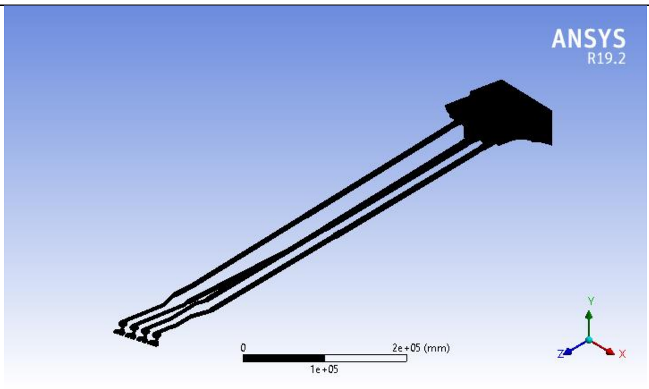
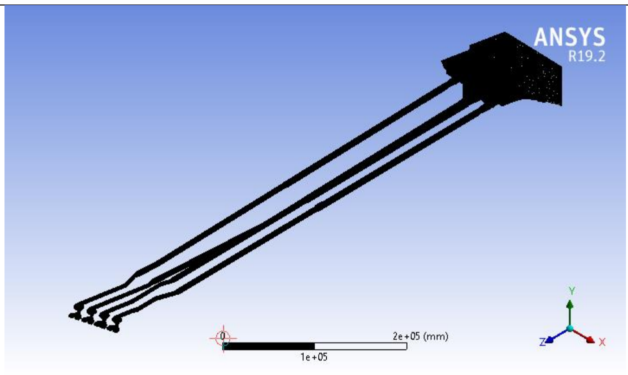
The processed data are then imported to the CAD software, SolidWorks. The CAD file of the dam is a combination of both topology surfaces and penstock drawing. And from here, the geometrical files are then exported into ANSYS workbench.

3.2.2 Meshing

The Temenggor hydropower dam water flow was simulated using Ansys Fluent. Figure below shows the CAD drawing of the dam. Like any other simulations, the meshing step is crucial as it influences the accuracy, convergences and computational load [32]. The simulation was done on 1 to 1 scale. Dealing with such big scale can be time consuming process which in turn can exhaust the computational resources.

Before performing the simulation, mesh sensitivity analysis needs to be performed first. Sensitivity analysis is performed by varying the mesh parameters and the simulation are done in order to compare the results similarity. Because of geometrical and floating-point issue, tetrahedron mesh will be used instead of hexahedral and cutcell. Discharge outlet will be used to compare the effect of different mesh variation, with tetrahedron mesh at different element size and at 50-time step with time step size 0.01 s. This setting was enough to observe the physical phenomenon.

Element type	Element size (mm)	Visuals

Tetrahedron	500	
	1000	
	1500	
	2000	

After running the test simulations, the results were plot and compared as in below.

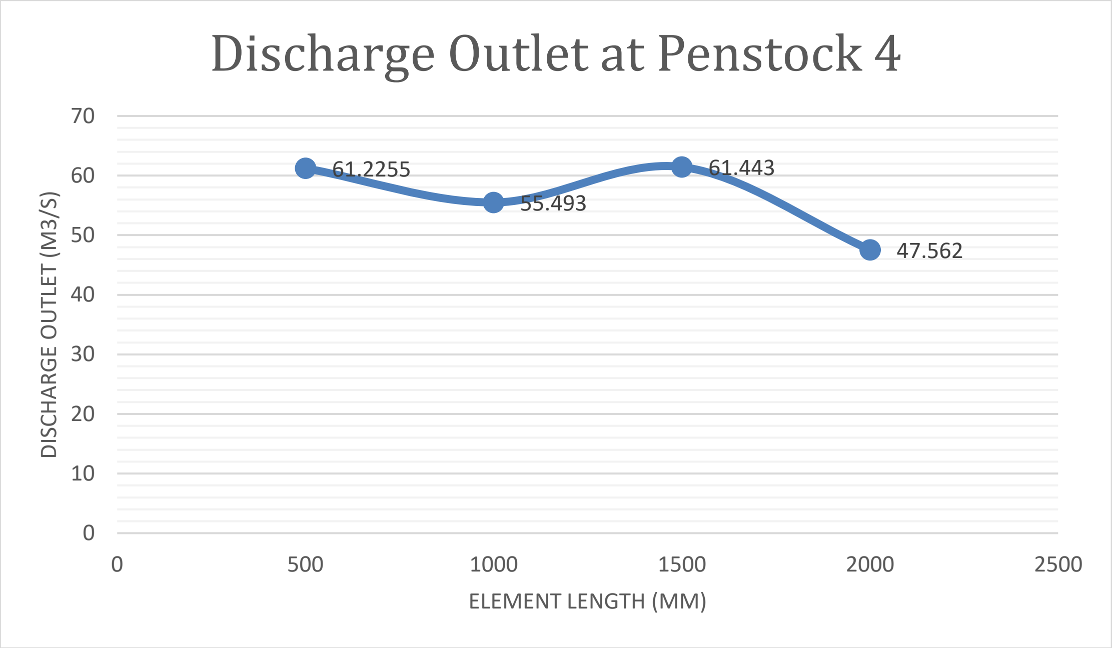


Figure 3.8 Grid independence test results from varying element length

From the mesh sensitivity analysis, element 1500 mm and 500 mm shows small difference with around 0.355% difference. And element length with 2000 mm shows the highest variation compared to the rest followed not far behind 1000 mm. So, the choices are between 500 mm and 1500 mm because the percentage difference is within 2%.

In the end, the dam was meshed using *tetrahedron assembly method with 1500 mm element size*. These mesh parameters can optimize the computational power while yielding good results accuracy.